Common Model Stability Problems When Performing an Unsteady Flow Analysis

Gary W. Brunner, P.E.

Hydrologic Engineering Center 1

Objectives

- For students to have a better understanding of model stability problems.
- To become familiar with the available parameters and techniques within HEC-RAS that will allow you to develop a stable and accurate model.
- To learn how to detect, find, and fix model stability problems.

Overview

- Model Accuracy and Stability
- Factors Affecting Model Stability
  - Cross section spacing
  - Computational time step selection
  - Theta weighting factor
  - Calculation tolerances and iterations
  - Lateral Structures/weirs
  - Manning’s n values
  - Initial/Low flow conditions
  - Steep Streams/Mixed Flow regime
  - Drops in the bed profile
  - Bridge/Culverts
  - Cross section geometry and table properties
  - Breach characteristics
- Detecting and fixing Stability Problems

Model Accuracy

- Accuracy can be defined as the degree of closeness of the numerical solution to the true solution.

- Accuracy depends upon the following:
  - Assumptions and limitations of the model (i.e. one dimensional model, single water surface, etc…)
  - Accuracy of the geometric Data (cross sections, Manning's n values, bridges, culverts, etc…)
  - Accuracy of the flow data and boundary conditions
  - Numerical Accuracy of the solution scheme

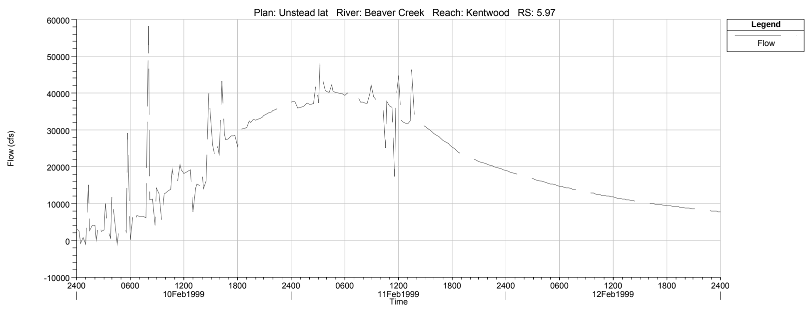
Hydrologic Engineering Center 4

Numerical Accuracy

- If we assume that the 1-dimensional unsteady flow equations are a true representation of flow moving through a river system. Then only an analytical solution of these equations will yield an exact solution.
- Finite difference solutions are approximate.
- An exact solution of the equations is not feasible for complex river systems, so HEC-RAS uses a finite difference scheme.

Model Stability

- An unstable numerical model is one for which certain types of numerical errors grow to the extent at which the solution begins to oscillate, or the errors become so large that the computations can not continue.

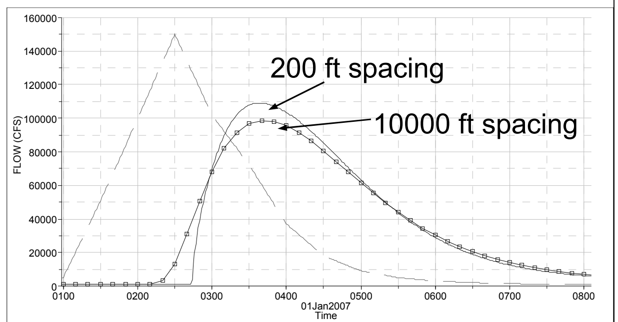


Hydrologic Engineering Center 6

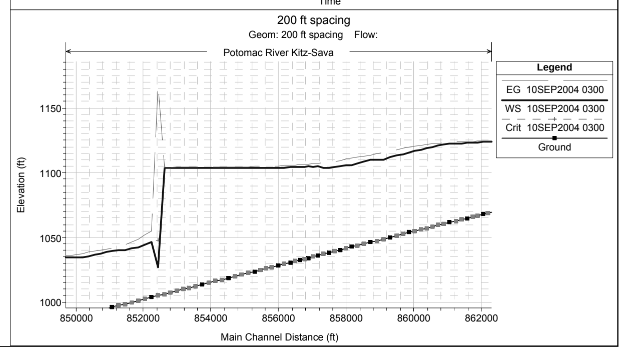
Developing a stable model is a common problem when working with an unsteady flow model of any size or complexity. Modeling a dam break flood wave is one of the most difficult unsteady flow problems to model.

Cross Section Spacing

- Cross sections placed to far apart can cause numerical damping of the flood wave (to low of a peak flow downstream), and/or model instability.



- Cross sections placed to close together can cause wave steepening and model instability on the rising side of the flood wave.



Hydrologic Engineering Center

Not enough cross sections: When cross sections are spaced far apart, and the changes in hydraulic properties are great, the solution can become unstable. In general, cross sections spaced too far apart will cause additional numerical diffusion, due to the derivatives with respect to distance being averaged over to long of a distance. Also, if the distance between cross sections is so great, such that the Courant number would be much greater than 1.0, then the model may also become unstable.

Cross Sections too Close. If the cross sections are too close together, then the derivatives with respect to distance may be overestimated, especially on the rising side of the flood wave. This can cause the leading edge of the flood wave to over steepen, to the point at which the model may become unstable.

XS Spacing
Maximum and Minimum

Use Dr. Fread’s and Samuals equations as a guide for maximum spacing.

Dr. Fread’s Equation:

$$\Delta x \leq \frac{cT_r}{20}$$

Samuals Equation:

$$\Delta x \leq \frac{0.15D}{S_0}$$

Where:

Where:	Δx	= Cross section spacing (feet)
	T_r	= Time of rise of the main flood wave (seconds)
	c	= Wave speed of the flood wave (ft/s)
	D	= Average bank full depth of the channel (ft)
	S_0	= Average bed slope (ft/ft).

Minimum spacing for a dam break model should be in the range of 50 to 100 ft.

Hydrologic Engineering Center 8

One of the first steps in stabilizing a dam breach model is to apply the correct cross section spacing. Fread’s equation and Samuel’s equations are good starting points. Samuels equation is a little easier to use since you only have to estimate the depth and slope. Frequently, bank full depth is used. For Fread’s equation, although the time of rise of the hydrograph (Tr) is easy enough to determine, the wave speed (c) is a little more difficult to come by. Once a cross section spacing is decided upon, apply it to the entire reach using the HEC-RAS cross section interpolation routines. Make sure that the reach-wide method is applicable. At areas of extreme contraction and expansion, at grade breaks, or in abnormally steep reaches, further localized interpolation may be necessary.

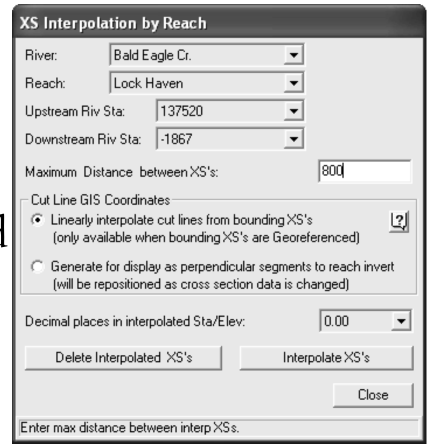
Fread, D.L. (1988) (Revision 1991). “The NWS DAMBRK Model. Theoretical Background and User’s Documentation.” National Weather Service, Office of Hydrology, Silver Spring, Md.

Fread, D.L., Lewis, J.M. (1993). “Selection of Dx and Dt Computational Steps for Four-Point Implicit Nonlinear Dynamic Routing Models” ASCE National Hydraulic Engineering Conference Proceedings, San Francisco, CA.

Samuels, P.G. (1989). “Backwater lengths in rivers”, Proceedings -- Institution of Civil Engineers, Part 2, Research and Theory, 87, 571-582.

Cross Section Interpolation

- Apply the XS interpolation to ensure a maximum spacing is not exceeded.
- At problem areas you may need to use tighter spacing:
  - Steep reaches
  - Transition zones
  - Grade breaks

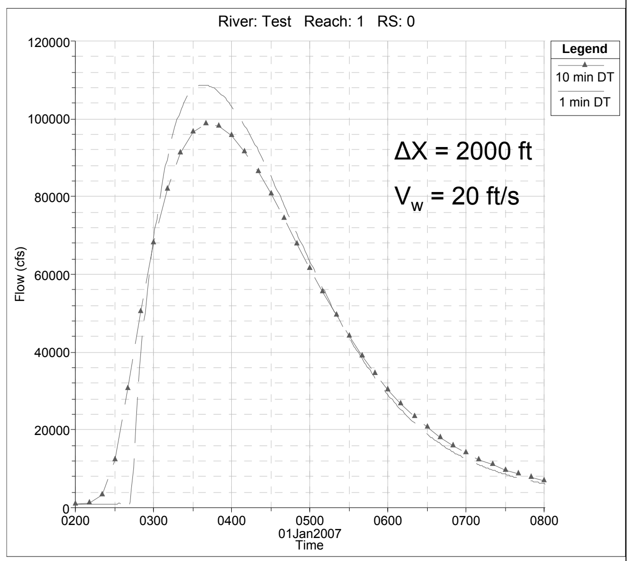


Hydrologic Engineering Center 9

In general, it is always better to use real cross sections rather than interpolated. However, if acquiring more cross section data is not possible, then the cross section interpolation routines in HEC-RAS should be used to ensure that the cross section do not go over a maximum distance estimated from Samual’s, or Fread’s equation.

Computational Time Step

- To large a time step will cause numerical diffusion (attenuation of the peak) and also model instability.
- To small of a time step can also lead to model instability as well as very long computation times.
- For this example a 5 sec time step caused the model to go unstable.



Hydrologic Engineering Center 10

Too large of a time step: When the solution scheme solves the unsteady flow equations, derivatives are calculated with respect to distance and time. If the changes in hydraulic properties at a give cross section are changing rapidly with respect to time, the program may go unstable. The solution to this problem in general is to decrease the time step.

Too Small of a Time Step. If a time step is selected that is much smaller than what the Courant condition would dictate for a given flood wave, this can also cause model stability problems. In general to small of a time step will cause the leading edge of the flood wave to steepen, possible to the point of oscillating and going unstable.

Computational Time Step - continued

Stability and accuracy can be achieved by selecting a time step that satisfies the Courant Condition :

$$C_r = V_w \frac{\Delta t}{\Delta x} \leq 1.0 \qquad \Delta t \leq \frac{\Delta x}{V_w}$$

For most rivers, the flood wave velocity is calculated more accurately by:

$$V_w = \frac{dQ}{dA}$$

An approximate flood wave velocity can be calculated as:

$$V_w = \frac{3}{2} \overline{V}$$

Hydrologic Engineering Center 11

Where: V_w = The flood wave speed, which is normally greater than the average velocity.

V = Average velocity of the flow

Δx = Distance between cross sections

Δt = computational time step

Q = flow rate

A = Flow area

User's should pay close attention to the Courant condition for selecting the computational interval.

Practical Time Step Selection

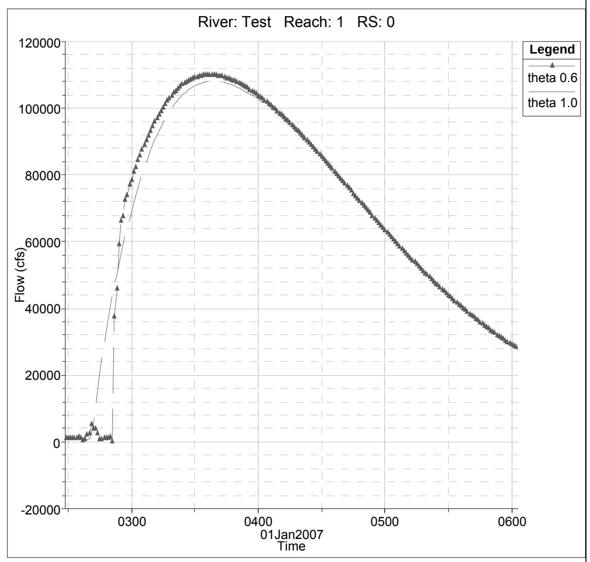
- For medium to large rivers the Courant condition may yield time steps that are to restrictive (i.e. a larger time step could be used and still maintain accuracy and stability).
- A practical time step is = $\Delta t \leq \frac{Tr}{20}$
- Remember that for Dambreak models, typical time steps are in the range of 1- 60 seconds due to the very fast flood wave velocities..

Hydrologic Engineering Center 12

Where: T_r = Time of rise of the flood wave.

Theta Weighting Factor

- Theta is a weighting applied to the finite difference approximations when solving the unsteady flow equations.
- Theoretically Theta can vary from 0.5 to 1.0. However a practical limit is from 0.6 to 1.0
- Theta of 1.0 provides the most stability. Theta of 0.6 provides the most accuracy.
- The default in HEC-RAS is 1.0. Once you have your model developed, try to reduce theta towards 0.6, as long as the model stays stable.



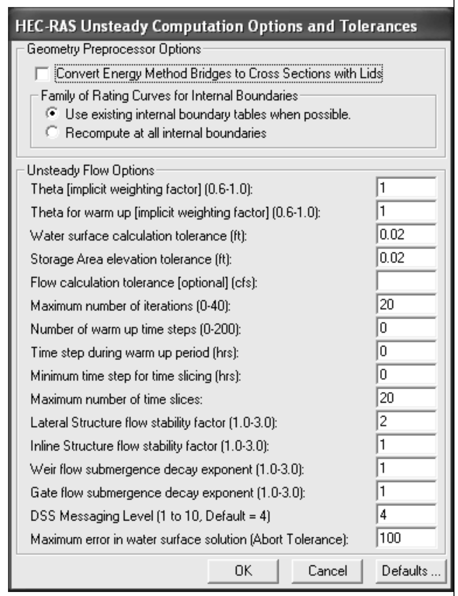
Hydrologic Engineering Center 13

Larger values of theta increase numerical diffusion, but, by how much? Experience has shown that for short period waves that rapidly rise, theta of 1.0 can produce significant errors. However, these errors can be reduced by using smaller time steps.

When choosing theta, one must balance accuracy and computational robustness. Larger values of theta produce a solution that is more robust, less prone to blowing up. Smaller values of theta, while more accurate, tend to cause oscillations in the solution, which are amplified if there are large numbers of internal boundary conditions.

Calculation Options and Tolerances

- Theta
- Water surface/SA/flow tolerance
- Maximum Number of Iterations
- Warm up: duration and time step
- Time slicing
- Lateral/Inline Structure Stability
- Weir/Gate flow submergence



Hydrologic Engineering Center 14

Water Surface, Storage Area, and Flow Tolerances: Three solution tolerances can be set or changed by the user: Water surface calculation (0.02 default); Storage area elevation (0.05 default); and Flow calculation (Default is that it is not used). The default values should be good for most river systems. Only change them if you are sure!!!

Making the tolerances larger can reduce the stability of the solution. Making them smaller can cause the program to go to the maximum number of iterations every time.

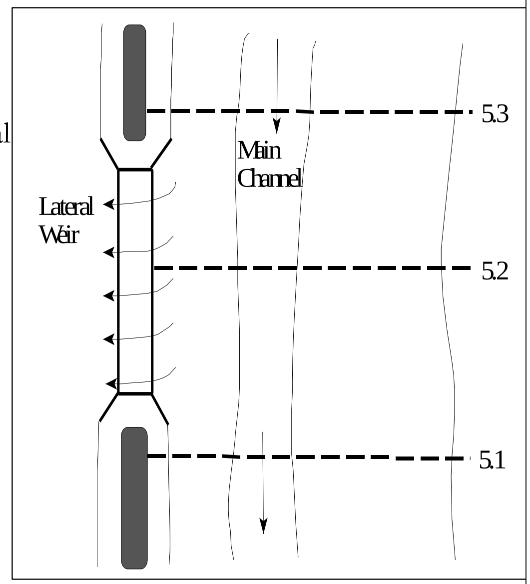
Maximum Number of Iterations: At each time step derivatives are estimated and the equations are solved. All of the computation nodes are then checked for numerical error. If the error is greater than the allowable tolerances, the program will iterate. The default number of iterations in HEC-RAS is set to 20. Iteration will generally improve the solution. This is especially true when your model has lateral weirs and storage areas.

Warm up time step and duration: The user can instruct the program to run a number of iterations at the beginning of the simulation in which all inflows are held constant. This is called the warm up period. The default is not to perform a warm up period, but the user can specify a number of time steps to use for the warm up period. The user can also specify a specific time step to use (default is to use the user selected computation interval). The warm up period does not advance the simulation in time, it is generally used to allow the unsteady flow equations to establish a stable flow and stage before proceeding with the computations.

Time Slicing: The user can control the maximum number of time slices and the minimum time step used during time slicing. There are two ways to invoke time slicing: rate of change of an inflow hydrograph or when a maximum number of iterations is reached.

Inline/Lateral Structure Stability Issues

- Long and flat lateral weirs can often be a source of model instability.
  - Small change in stage in the river results in big change in flow going out the lateral structure
  - Flow is assumed to be constant over the time step
- Solutions:
  - Reduce the computation interval
  - Put a small slope on the lateral weir
  - Use lateral structure stability factors
- Opening and Closing Gates Quickly
  - Reduce the computational time step
  - Open and close gates slower



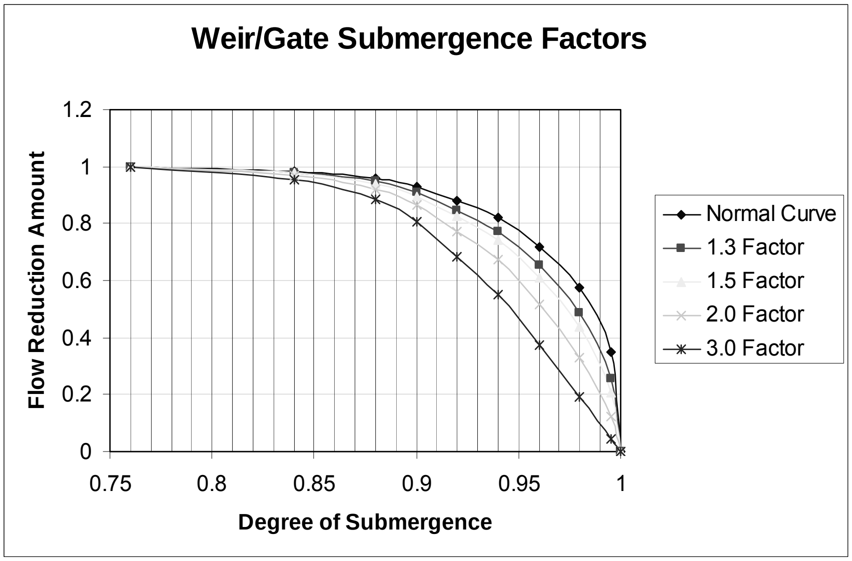
Hydrologic Engineering Center 15

Inline and Lateral Structures can often be a source of instability in the solution. Especially lateral structures, which take flow away or bring it into the main river. During each time step, the flow over a weir/spillway is assumed to be constant. This can cause oscillations by sending too much flow during a time step. One solution is to reduce the time step. Another solution is to use Inline and Lateral Structure stability factors, which can smooth these oscillations by damping the computed flows. However, using these stability factors can reduce the accuracy of the computed values. The Inline and Lateral Structure stability factors can range from 1.0 to 3.0. The default value of 1.0 is essentially no damping of the computed flows. As you increase the factor you get greater dampening of the flows (which will provide for greater stability), but less accuracy.

Long and flat Lateral Weirs/Spillways: during the computations there will be a point at which for one time step no flow is going over the lateral weir, and then the very next time step there is. If the water surface is rising rapidly, and the weir is wide and flat, the first time the water surface goes above the weir could result in a very large flow being computed (i.e. it does not take a large depth above the weir to produce are large flow if it is very wide and flat). This can result in a great decrease in stage from the main river, which in turn causes the solution to oscillate and possible go unstable. This is also a common problem when having large flat weirs between storage areas. The solution to this problem is to use smaller computational time steps, and/or weir/spillway stability factors.

Opening gated spillways to quickly: When you have a gated structure in the system, and you open it quickly, if the flow coming out of that structure is a significant percentage of the flow in the receiving body of water, then the resulting stage, area and velocity will increase very quickly. This abrupt change in the hydraulic properties can lead to instabilities in the solution. To solve this problem you should use smaller computational time steps, or open the gate a littler slower, or both if necessary.

Weir/Gate Submergence Exponents

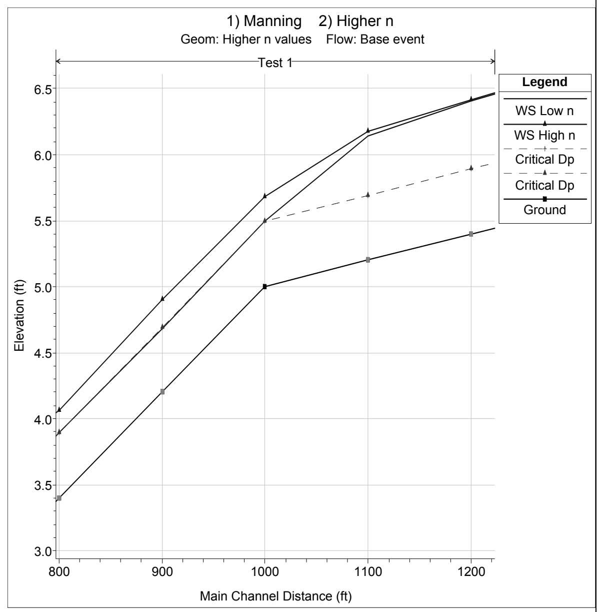


Hydrologic Engineering Center 16

To reduce the oscillations, the user can increase the Weir/Gate Submergence decay exponent. This factor can vary from 1.0 to 3.0. A factor of 1.0 leaves the submergence criteria in its original form. Using a factor greater than one causes the program to use larger submergence factors earlier, and makes the submergence curve less steep at high degrees of submergence. A plot of the submergence curves for various factors is shown in the Figure above.

Manning's n Values

- Manning's n values that are too low can cause model instability
  - Lower depths
  - Higher velocities
  - Supercritical flow
  - Flow transitions
- Manning's n values that are too high will locally increase stage and attenuate the hydrograph more as it moves downstream



Hydrologic Engineering Center 17

Manning's n values can also be a source of model instability. Manning's n values that are too low, will cause shallower depths of water, higher velocities, and possibly even supercritical flows. This is especially critical in steep streams, where the velocities will already be high. User's should check there estimated Manning's n values closely in order to ensure reasonable values. It is very common to underestimate Manning's n values in steep streams. Use Dr. Robert Jarrets equation for steep streams to check your main channel Manning's n values.

Over estimating Manning's n values will cause higher stages and more hydrograph attenuation than may be realistic.

Low Flow Conditions

- Low flows are often a source of model instability
  - Pools and riffles (flow passes through critical depth)
  - Very shallow depths (When flood wave starts the change in depth/velocity is very large, therefore derivatives are large)
- Solutions
  - Increase Base Flow of Hydrograph (change hydrographs directly or use the Qmin option on the hydrograph editor)
  - Rule of thumb: Start with 1% of peak, don't exceed 10% of peak
  - Use a "Pilot Channel" to smooth our bed irregularities and provide some artificial depth.

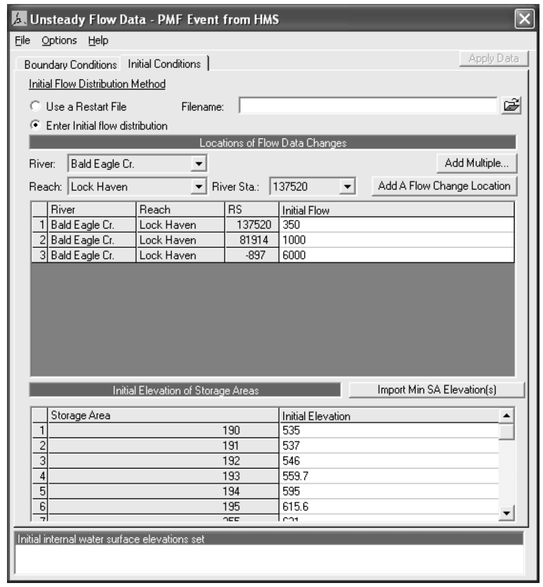
Hydrologic Engineering Center 18

If any portion of an inflow hydrograph is so low that it causes the stream to go through a pool and riffle sequence, it may be necessary to increase the base flow. The minimum flow value must be small enough that it is negligible when compared to the peak of the flood wave. A good rule of thumb is to start with a minimum flow equal to about 1 % of the peak flood (inflow hydrograph, or dam breach flood wave) and increase as necessary to 10%. If more than 10% is needed, then the problem is probably from something else.

Very shallow depths of water: When starting a simulation it is very common to start the system at low flows. If you have some cross sections that are fairly wide, the depth will be very small. As flow begins to come into the river, the water surface will change quickly. The leading edge of the flood wave will have a very steep slope. Sometimes this steep slope will cause the solution to reduce the depth even further downstream of the rise in the water surface, possible even producing a negative depth. This is do to the fact that the steep slope gets projected to the next cross section downstream when trying to solve for its water surface. The best solution to this problem is to use what is called a pilot channel. A pilot channel is a small slot at the bottom of the cross section, which gives the cross section a greater depth, without adding much flow area. This allows the program to compute shallow depths on the leading edge of the flood wave without going unstable. Another solution to this problem is to use a larger base flow at the beginning of the simulation.

Initial Conditions

- Initial condition flows need to be consistent with boundary condition flows at time zero.
  - If dendritic system – can leave initial condition flows blank.
- Initial Reservoir elevations need to be consistent with initial flows and gate settings used.
- Too low of, or inconsistent, initial conditions flow can often cause the model to blow up right at the start of the simulation



Hydrologic Engineering Center 19

Make sure that the initial conditions flow is consistent with the first time step flow, or minimum flow value, which ever is greater. User's must also pay close attention to initial gate settings for the reservoir, and the initial stage of the pool in the reservoir. The initial condition flow values must be consistent with all inflow hydrographs, as well as the initial flows coming out of the reservoir.

Flows entered on the initial conditions tab are used for calculating stages in the river system based on steady flow backwater calculations. If these flows and stages are inconsistent with the initial flows in the hydrographs, and coming out of the reservoir, then the model may have computational stability problems at the very beginning of the unsteady flow computations.

Steep Streams and Mixed Flow Regime

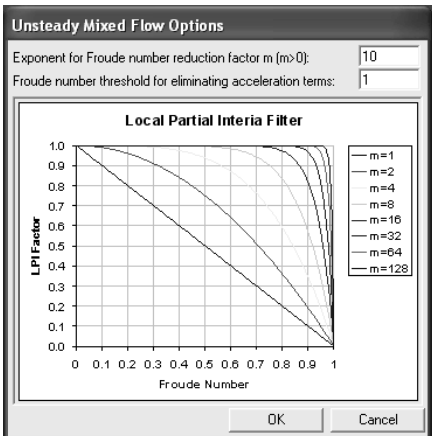
- Higher velocities and rapid changes in depth and velocity are more difficult to model and keep a stable solution.
- As Froude number approaches 1.0 (critical depth), the inertial terms of the St. Venant equations and their derivatives tend to cause model instabilities.
- Model goes to critical depth – RAS is limited to subcritcal flow for unsteady flow simulations, unless you turn on the mixed flow option.
- Solutions:
  - Higher n values (n values often under estimated in steep streams)
  - Increase base flow of hydrographs and initial condition flows
  - Try turning on the Mixed Flow regime option

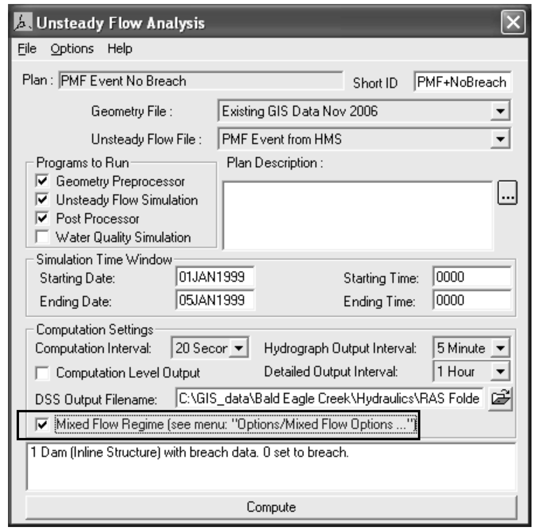
Hydrologic Engineering Center 20

Model goes to critical depth: The default solution methodology for unsteady flow routing within HEC-RAS is generally for subcritcal flow. However, the software does have an option to run in a mixed flow regime mode. However, this option should not be used unless you truly believe you have a mixed flow regime river system. If you are running the software in the default mode (subcritical only, no mixed flow), and if the program goes down to critical depth at a cross section, the changes in area, depth, and velocity are very high. This sharp increase in the water surface slope will often cause the program to overestimate the depth at the next cross section upstream, and possible underestimate the depth at the next cross section downstream (or even the one that went to critical depth the previous time step). One solution to this problem is to increase the Manning's n value in the area where the program is first going to critical depth. This will force the solution to a subcritical answer and allow it to continue with the run. If you feel that the true water surface should go to critical depth, or even to a supercritical flow regime, then the mixed flow regime option should be turned on. Another solution is to increase the base flow in the hydrographs, as well as the base flows used for computing the initial conditions. Increased base flow will often dampen out any water surfaces going towards or through critical depth due to low flows.

Mixed Flow Regime Option for Unsteady Flow

- HEC-RAS uses the Local Partial Inertia technique to model mixed flow regimes.
- User's can adjust the LPI parameters





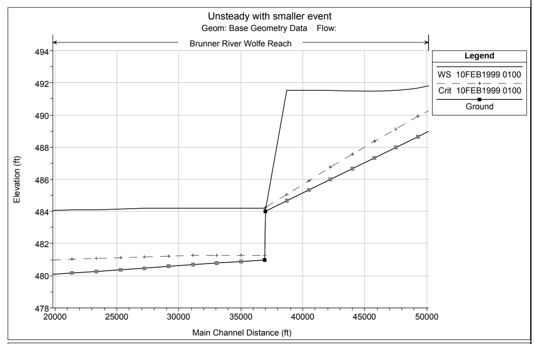
Hydrologic Engineering Center 21

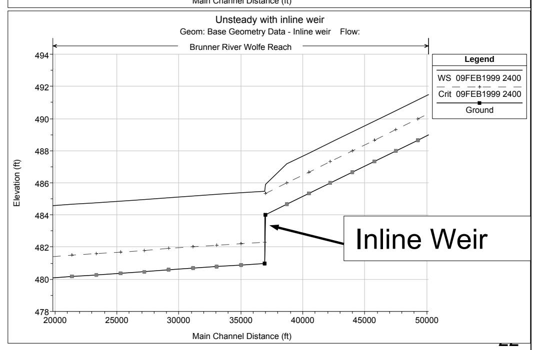
In order to solve the stability problem for a mixed flow regime system, Dr. Danny Fread (Fread, 1986) developed a methodology called the "Local Partial Inertia Technique." The LPI method has been adapted to HEC-RAS as an option for solving mixed flow regime problems when using the unsteady flow analysis portion of HEC-RAS. This methodology applies a reduction factor to the two inertia terms in the momentum equation as the Froude number goes towards 1.0.

The default values for the equation are FT = 1.0 (Froude Number Threshold) and m = 10 (exponent). When the Froude number is greater than the threshold value, the factor is set to zero. The user can change both the Froude number threshold and the exponent. As you increase the value of both the threshold and the exponent, you decrease stability but increase accuracy. As you decrease the value of the threshold and/or the exponent, you increase stability but decrease accuracy. To change either the threshold or the exponent, select **Mixed Flow Options** from the **Options** menu of the Unsteady Flow Analysis window.

Drops In The Bed Profile

- Significant drops in the elevation of the channel bed can cause flow to pass through critical depth and results in an unstable model solution.
- Solutions:
  - Use an Inline Weir to model drop
  - Increase Base Flow
  - more cross sections and turn on mixed flow regime option
  - Rating curve for cross section at top of the drop





Hydrologic Engineering Center

Significant drops in the bed profile can also be a source of model stability problems, especially at low flows. If the drop is very small, then usually an increase in baseflow will drowned out the drop, thus preventing the model from passing through critical depth. If the drop is significant, then it should be modeled with an inline structure using a weir. This will allow the model to use a weir equation for calculating the upstream water surface for a given flow, rather than using the unsteady flow equations. This produces a much more stable model, as the program does not have to model the flow passing through critical depth with the unsteady flow equations. HEC-RAS automatically handles submergence on the weir, so this is not a problem.

Bridges and Culverts

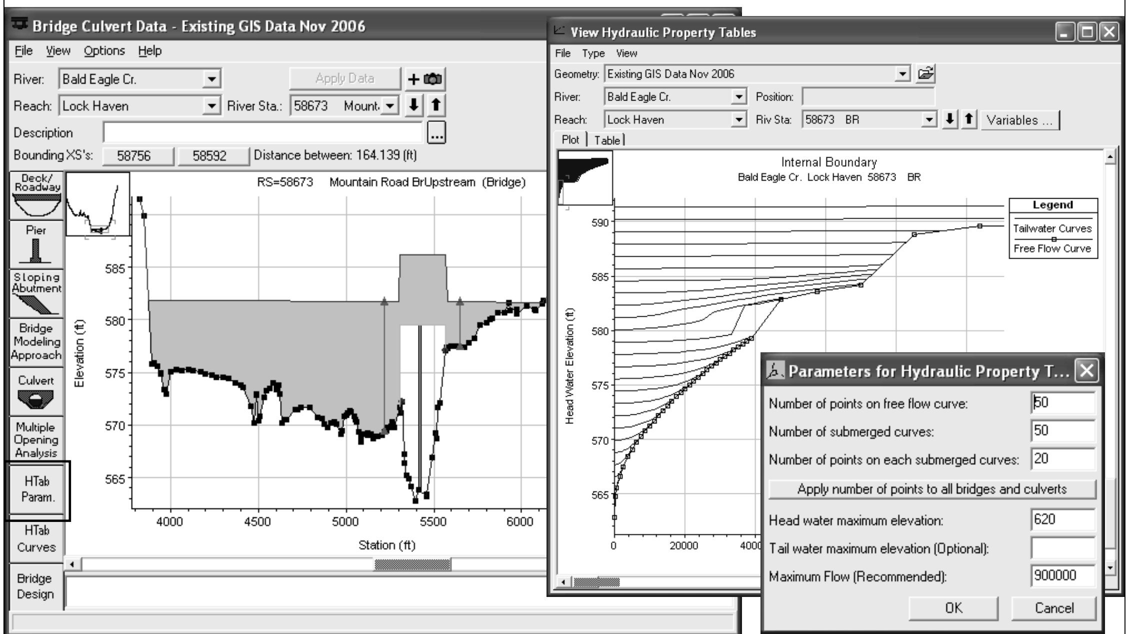
- It is very common to have rapid changes in depth and velocity at Bridges and Culverts, however this can be a source of model instability
- As flow transitions from low flow to pressure flow at the structure the water surface upstream will jump up very quickly.
- HEC-RAS pre-processes bridges/culverts into a family of curves (tailwater/headwater vs flow). Common Problems with the curves are:
  - Curve extents not high enough (change default extents)
  - Abrupt transitions in curves (adjust bridge parameters or use smaller ΔT)

Hydrologic Engineering Center 23

Bridge/Culvert crossings can be a common source of model stability problems when performing a Dam Break analysis. Many bridges will be overtopped during such an event. Many of those bridges may in fact be washed out during such an event. Common problems at bridges/culverts are the extreme rapid rise in stages when flow hits the low chord of the bridge deck or the top of the culvert. Modelers need to check the computed curves closely and make sure they are reasonable. One solution to this problem is to use smaller time steps, such that the rate of rise in the water surface is smaller for a given time step. Modelers may also need to change hydraulic coefficients to get curves that have more reasonable transitions.

An additional problem is when the curves do not go high enough, and the program extrapolates from the last two points in the curve. This extrapolation can cause problems when it is not consistent with the cross section geometry upstream and downstream of the structure.

Bridge/Culvert Family of Curves

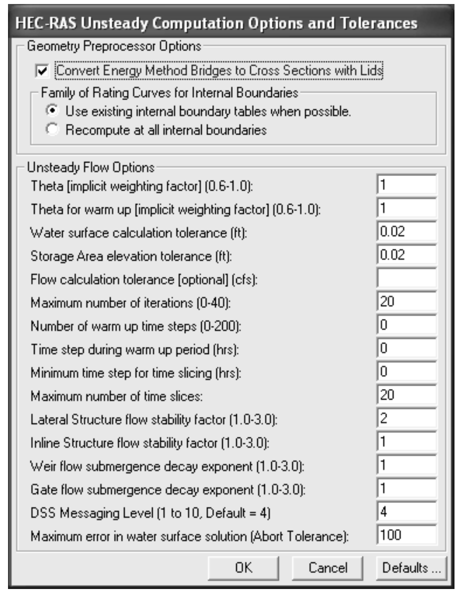


Hydrologic Engineering Center 24

Bridge/Culvert Family of rating curves: The program creates a family of rating curves to define all the possible headwater, tailwater, and flow combinations that can occur at a particular structure. The user can control how many submerged curves get calculated (default = 50), how many points in each curve (default = 20), and the properties used to define the limits of the curves (maximum headwater, maximum tailwater, maximum flow, and maximum head difference). By default, the software will take the curves up to an elevation equal to the highest point in the cross section just upstream of the structure. This may lead to curves that are too spread out and go up to a flow rate that is way beyond anything realistic for that structure. These type of problems can be reduced by putting in specific table limits for maximum headwater, tailwater, flow, and head difference.

Convert Energy Bridges to Cross Sections with Lids

- HEC-RAS has on option to model a bridge as normal cross sections with a lid instead of the family of curves.
  - Good option when bridge/embankment are a small obstruction to flow
  - Bad idea when depth/velocity will change rapidly through the bridge (Will often cause model to blow up)

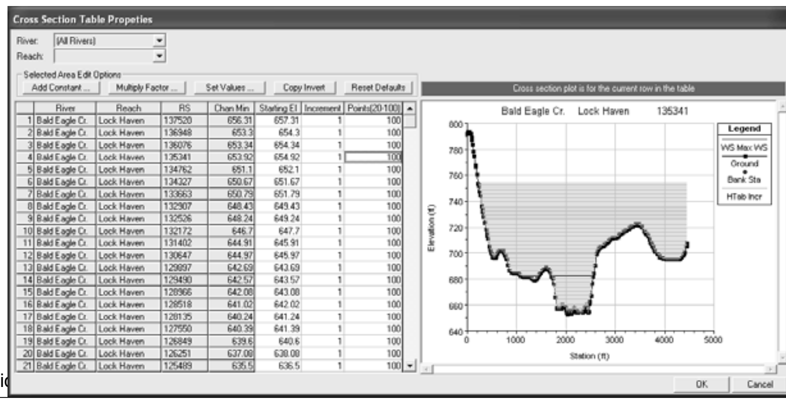


Hydrologic Engineering Center 25

The option to convert bridges that have been modeled with the energy equation for low and high flow is a good option if the structure and the road embankment are a small obstruction to the flow. If the flow will have to contract greatly inside of the structure, then modeling the bridge in this manner may lead to an unstable modeling solution through the structure. If you have bridges and culverts that will cause the flow to contract greatly through the structure, and even if you have chosen to use the energy equation for low and high flow, computing a family of curves for the structure will produce a more stable model. When the family of curves is used, the program does not solve the momentum and continuity equations inside of the structure, only outside of the structure. The curves themselves are use to obtain a resulting headwater for a given flow and tailwater, without solving for the hydraulics inside of the structure.

Cross Section Geometry and Table Properties

- Bad cross section properties, commonly caused by: levee options, ineffective flow areas, Manning’s n values, etc…
- Cross section properties that do not go high enough, or are way too high (curves are spread to far apart).
- Not enough definition in the properties tables.



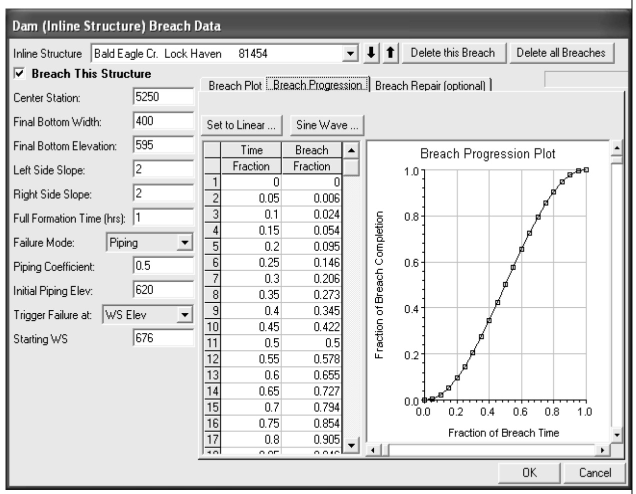
Bad cross section properties: All of the cross sections get converted to tables of hydraulic properties (elevation versus area, conveyance, and storage). If the curves that represent these hydraulic properties have abrupt changes with small changes in elevation, this can also lead to instability problems. This situation is commonly caused by: levees being overtopped with large areas behind them (since the model is one dimensional, it assumes that the water surface is the same all the way across the entire cross section); and ineffective flow areas with large amounts of storage areas that are turned on at one elevation, and then turn off at a slightly higher elevation (this makes the entire area now used as active conveyance area). There are many possible solutions to these problems, but the basic solution is to not allow the hydraulic properties of a cross section to change so abruptly. If you have a levee with are large amount of area behind it, model the area behind the levee separately from the cross section. This can be done with either a storage area or another routing reach, whichever is most hydraulically correct for the flow going over the levee or if the levee breaches. With large ineffective flow areas, the possible solutions are to model them as being permanently on, or to put very high Manning’s n values in the ineffective zones.

Cross section property tables that do not go high enough: The program creates tables of elevation versus area, conveyance, and storage area for each of the cross sections. These tables are used during the unsteady flow solution to make the calculations much faster. By default, the program will create tables that extend up to the highest point in the cross section, however the user can override this and specify their own table properties (increment and number of points). If during the solution the water surface goes above the highest elevation in the table, the program simply extrapolates the hydraulic properties from the last two points in the table. This can lead to bad water surface elevations or even instabilities in the solution.

Not enough definition in cross section property tables: The counter problem to the previous paragraph is when the cross section properties in a given table are spread too far apart, and do not adequately define the changes in the hydraulic properties. Because the program uses straight-line interpolation between the points, this can lead to inaccurate solutions or even instabilities. To reduce this problem, we have increased the allowable number of points in the tables to 100.

Breach Characteristics

- By default HEC-RAS uses a linear growth rate for forming the breach.
- Rapid change in flow at the beginning of the breach can cause instability
- Try non-linear breach to smooth out change in flow (Sine Wave or User entered values)



Hydrologic Engineering Center 27

If the user puts in a very large breach, over a very short period of time, and they use the linear breach growth rate, the model will have a very abrupt change in flow starting right at the beginning of the breaching process. This rapid change in flow at the leading edge of the flood wave may cause an instability at the beginning of the breaching process, just downstream of the dam. Some possible solutions to this are:

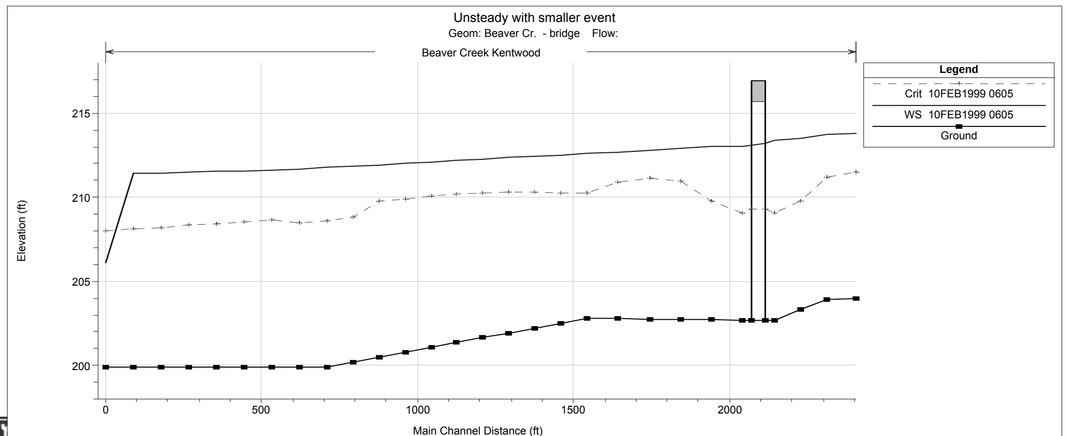
•Smaller time step

•Use the non-linear breach progression (sine wave or user entered)

•Increasing the overall breach time.

Bad Downstream Boundary Conditions

- Rating curve with bad points or not high enough
- Slope for normal depth to steep



Hydrologic Engineering Center 28

Bad downstream boundary condition: If the user entered downstream boundary condition causes abrupt jumps in the water surface, or water surface elevations that are too low (approaching or going below critical depth), this can cause oscillations in the solution that may lead to it going unstable and stopping. Examples of this are rating curves with not enough points or just simply to low of stages for a given flow; and normal depth boundaries where the user has entered to steep of a slope for the energy gradeline.

Detecting Stability Problems

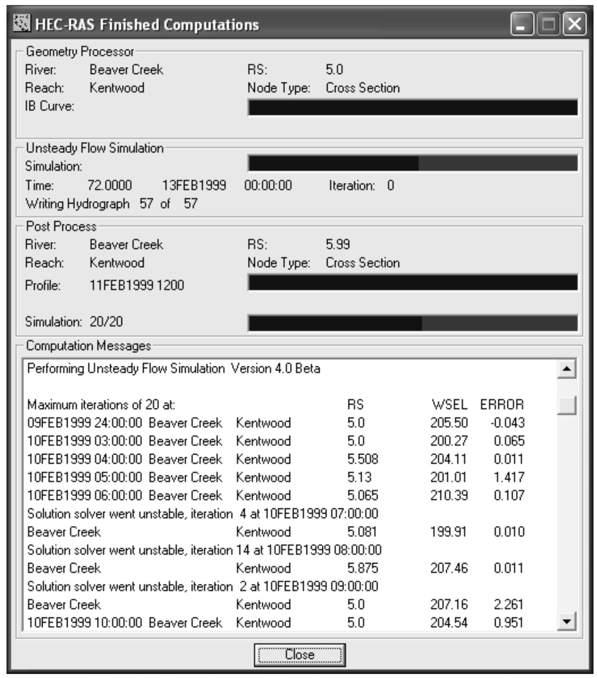
- How do you know you have a model stability problem?
  - Program completely blows up during run.
  - Program says matrix solution went completely unstable during the calculations.
  - Computed error in water surface calc is very large
  - Program goes to maximum number of iterations for several time steps in a row, with large errors.
  - Program has oscillations in the computed stage and flow hydrographs.

Detecting Stability Problems - Continued

- What do you do when this happens?
  - Note the simulation time and location from the computation window when the program either blew up or first started to go to the maximum number of iterations with large water surface errors.
  - Use the HEC-RAS Profile and Cross Section Plots as well as the Tabular Output to find the problem location and issue.
  - If you can not find the problem using the normal HEC-RAS output - Turn on the "Detailed Output for Debugging" option and re-run the program.
  - View the text file that contains the detailed log output of the computations. Locate the simulation output at the simulation time when the solution first started to go bad.
  - Find the river station locations that did not meet the solution tolerances. Then check the data in this general area.

Computation Window

- First place to look for problems
- When the maximum number of iterations is reach, and solution error is greater than the predefined tolerance, the time step, river, reach, river station, water surface elevation and the amount of error is reported.
- When the error increases too much, the solution will stop and say “Matrix Solution Failed”.
- Often the first RS to show up on the window can give clues to the source of instabilities

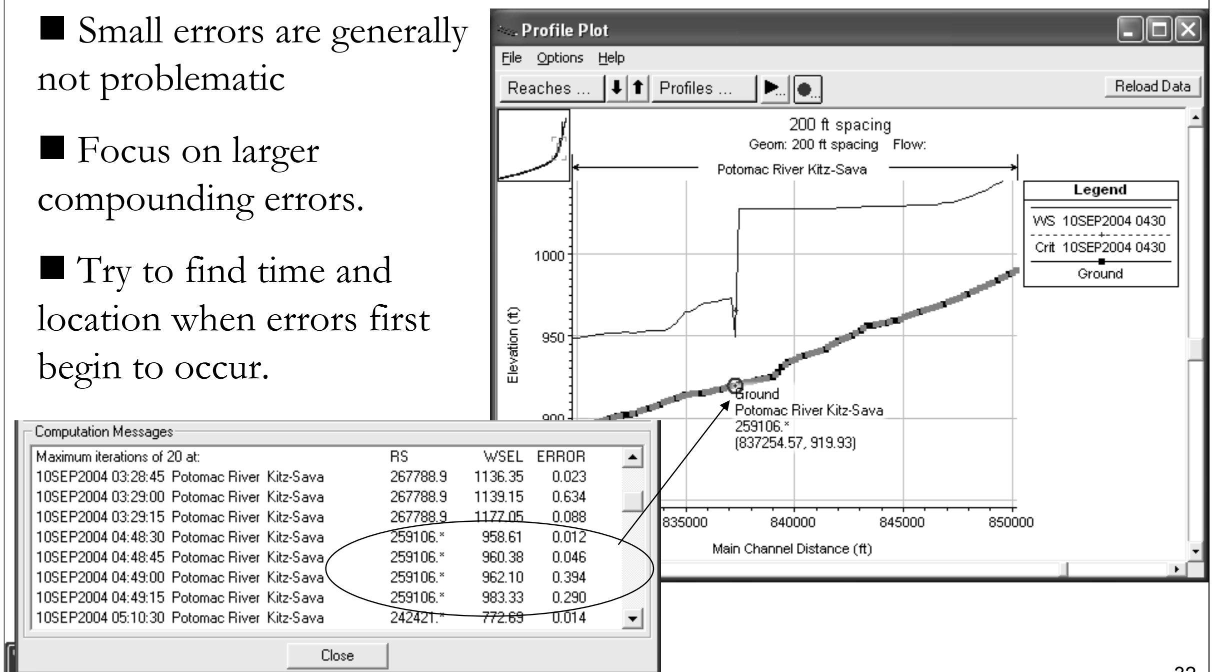


Hydrologic Engineering Center 31

The first place to look for instabilities and errors is the Computations Window during and just after the simulation is run. The red progress bar indicates the model went unstable and could not complete the simulation. The Computation Messages window provides a running dialog of what is happening in the simulation at a given time step in a given location. This allows the user to watch errors propagate during the simulation. Once the simulation has crashed, don’t close the Computations Window. Instead, scroll up through the messages and try to determine where the propagation of errors began, and at what time.

Computation Window

- Small errors are generally not problematic
- Focus on larger compounding errors.
- Try to find time and location when errors first begin to occur.



Hydrologic Engineering Center

32

Sometimes the first error to occur is at the beginning of the simulation and is just a result of the model settling out after the transition from initial conditions to the first time step. Particularly if the error only occurs once for that given river station. It is better to focus on reoccurring errors or compounding errors first. The example on this slide shows a relatively small error at river station 259106* that grows to 0.4 ft in the next few time steps.

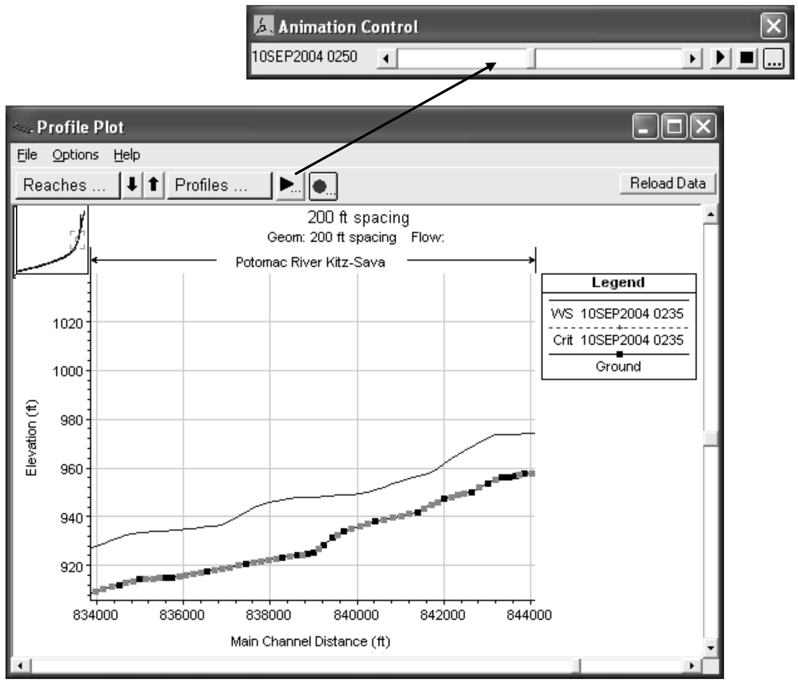
Profile Plot

- Great visual tool for finding problem areas.
- Use the "Animation" option to look for obvious instabilities. Zoom in to get a closer look.
- May need to refine the Detailed Output Interval to see where and when the instability occurs.
- When the first hints of an instability is revealed, click on that "node" and investigate further.

Hydrologic Engineering Center 33

The profile plot is typically the first graphical tool to use to try to pinpoint instabilities. Obvious errors are shown distinctly in this plot and you can see what is going on in the entire reach at the same time. Stepping through each profile using the animation tool allows you to see changes over time, including the progression of the flood wave as well as propagation of errors. The profile output is taken from the detailed output file. Therefore, it is sometimes necessary to refine the detailed output interval to adequately see the beginning of instabilities. The profile plot allows the user to click on a given node to determine its river stationing. Find the node where the instability first occurs and investigate further.

Profile Plot Animation

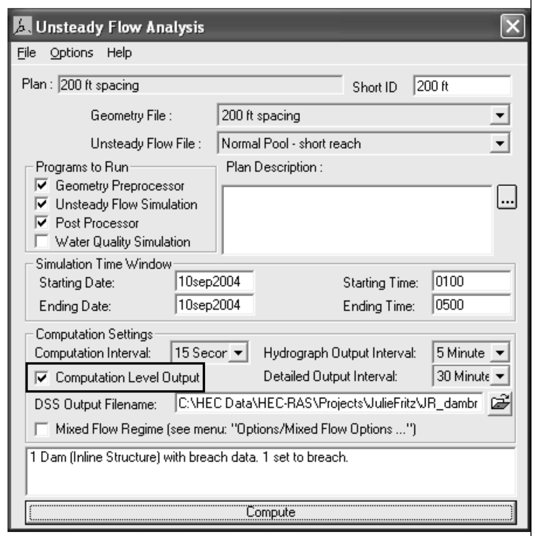


Hydrologic Engineering Center 34

The above slide was an animation of the profile plot, showing the progression of a model instability problem. The profile plot can assist you in locating where an instability is occurring and when. You may need to zoom in to get a closer look. You may also need to set the **Detailed Output Interval** to a smaller value and re-run the simulation in order to see what is happening at a finer time step increment.

Computation Level Output

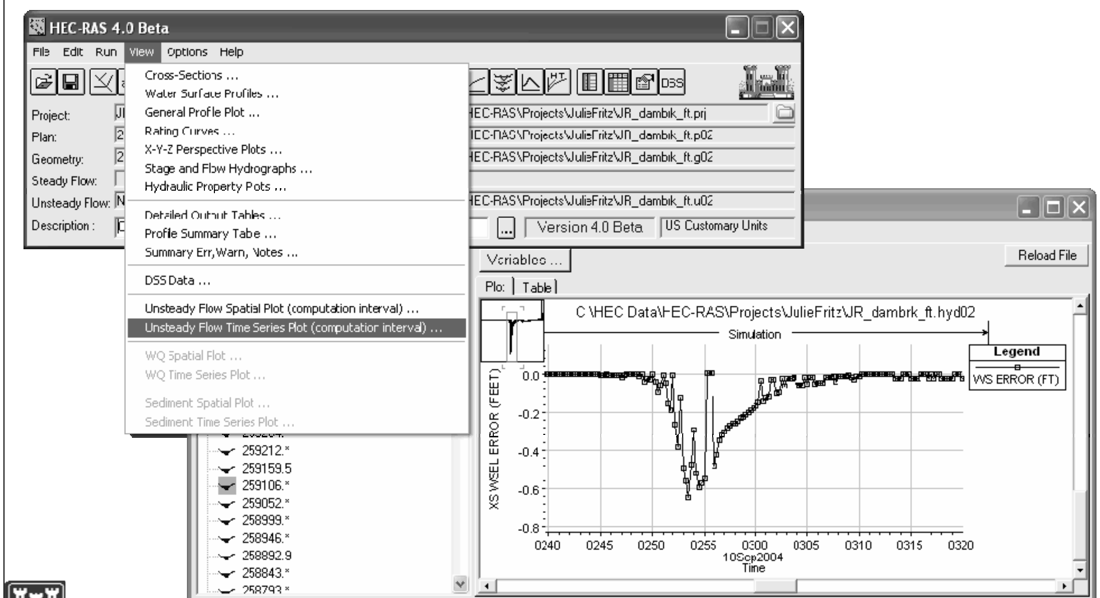
- Writes flow and stage at all locations to a separate file.
- Tools available from the View menu:
  - Spatial Plots
    - profile
    - schematic
  - Time Series plots
    - water surface, depth, flow
    - WS and flow errors
- Warning: Can create large output files when used with large data sets for long times



Hydrologic Engineering Center 35

When performing an unsteady flow analysis the user can optionally turn on the ability to view output at the computation interval level. This is accomplished by checking the box labeled **Computation Level Output** on the Unsteady Flow Analysis window (In the Computations Settings area on the window). When this option is selected an additional binary file containing output at the computation interval is written out. After the simulation the user can view computation level output by selecting either **Unsteady Flow Spatial Plot** or **Unsteady Flow Time Series Plot** from the **View** menu of the main HEC-RAS window.

Computation Level Output Visualization Tools

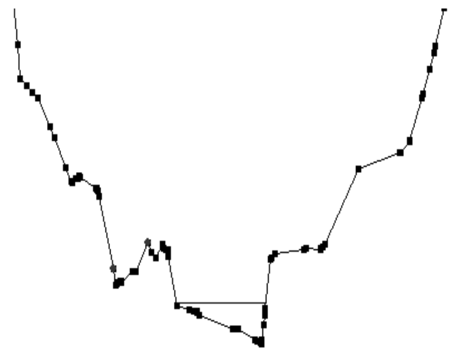


Visualization of computation level output can be accomplished with either **Spatial Plots** or **Time Series Plots**. From the Spatial Plots the user can view either a profile plot, a spatial plot of the schematic, or tabular output. The user can select from a limited list of variables that are available at the computation level output. These are water surface elevation (XS WSEl); Flow (XS Flow); computed maximum error in the water surface elevation (XS WSEL ERROR); computed maximum error in the flow (XS FLOW ERROR); and maximum depth of water in the channel (DEPTH). Each of the plots can be animated in time by using the video player buttons at the top right of the window. This type of output can often be very useful in debugging problems within an unsteady flow run. Especially plotting the water surface error and animating it in time.

The other type of plot available at the computation interval output level is the **Unsteady Flow Time Series Plot.** When this option is selected the user will get a plot as shown in the Figure above. Some of the same options and variables are available for the Time Series Plots as were available for the Spatial Plots.

Cross Section Plot

- Can help spot isolated problems such as:
  - Incorrect Bank Station locations
  - Bad Manning's n Values
  - Bad Station-elevation points
- Can help spot transition problems
  - Contraction/Expansion Areas
  - Ineffective Flow Areas
  - Levees

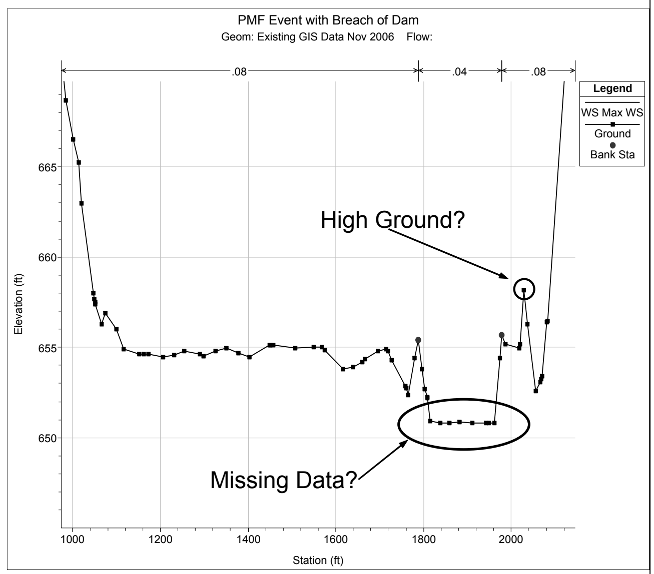


Hydrologic Engineering Center 37

Once a location for an instability is determined on the profile plot, the cross section plot can be used to investigate the cause of the instability. The cross section plot will show isolated problems such as incorrectly placed bank stations, poor n-values, and bad station-elevation data. In addition, scrolling through its neighboring cross sections can give you an idea of transition problems like contractions and expansions that occur to abruptly, poorly defined ineffective flow areas, or incorrectly handled levees or natural high ground spots.

Cross Section Plot

- Wide, Horizontal Beds
  - Estimated XS?
  - LIDAR, no bathymetry?
  - Prone to instabilities – High Area:Depth ratio
- High Ground
  - Levee Option
  - Ineffective Flows?
- Solutions?



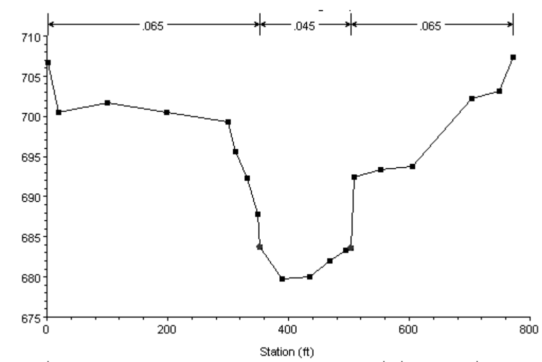
Hydrologic Engineering Center 38

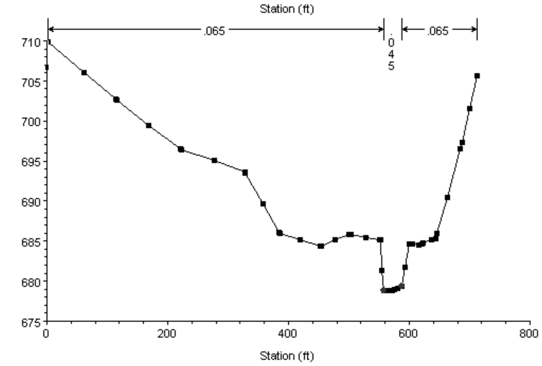
Another typical source of instabilities occurs when the main channel has a wide flat bed. This is usually found when cross sections are approximated or when terrain data is used to develop cross sections exclusive of real bathymetric data. Many times reaches are developed in GIS using LIDAR data or other aerial means. These survey methods don't penetrate water surfaces so the main channel is left with a flat horizontal bed equal to the water surface elevation. For shallow streams in dam breach analyses, this is normally okay, since the dam break flood wave is usually much greater than the depth of water. However, wide flat stream beds lend to instabilities because at lower flows, the area to depth ratio is very high. Again this presents the same problem of a small increase in depth amounting to a large relative increase.

Additionally, in the cross section plot, high ground that is not appropriately accounted for can be detected and fixed to remove sources of instabilities. High ground can be modeled as levees or with ineffective flows to remove the abrupt changes in storage and conveyance when the high ground is overtopped.

Cross Section Plot

- Transitions
  - If sudden contraction or expansion occurs over a short distance, how can this be handled?
  - Ineffective Flow Areas
  - More Cross Sections
    - Interpolation



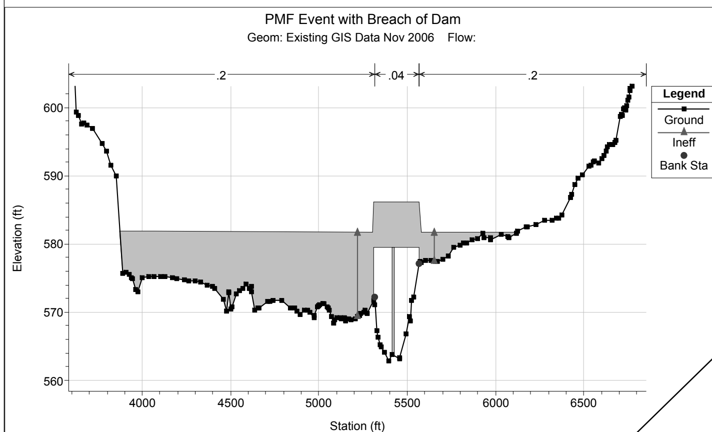


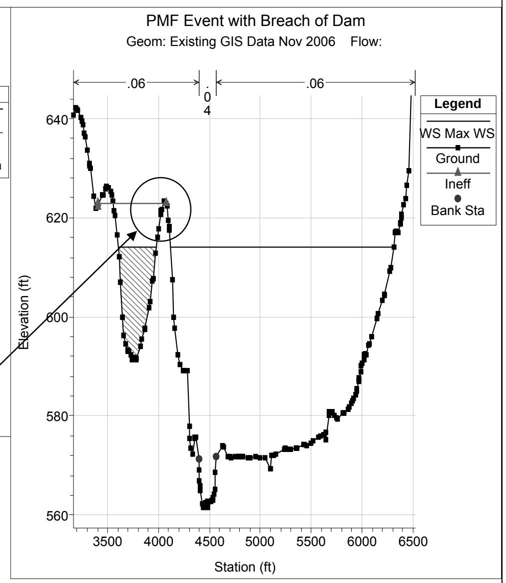
Hydrologic Engineering Center 39

The example in this slide shows an abrupt transition from a wide main channel to a narrow main channel. If these cross sections are close enough, the flow man not be able to contract so suddenly and the approximate numerical methods may not be able to handle this situation. In this case, ineffective flow areas can be placed in the wide cross section to help smooth the transition from wide to narrow. If these cross sections are far enough apart, then perhaps additional interpolated cross sections are warranted.

Cross Section Plot

- Ineffective Flow Areas





- Levee or Ineffective Flow? Or Neither?

Hydrologic Engineering Center

40

Ineffective flow areas are required up and downstream of bridges and culverts to properly define the contraction and expansion zones. Unsteady flow models, and particularly dam breach models, need these zones to be adequately defined. When the bridge is overtopped, the ineffective flow areas will turn off. This sudden and large increase in conveyance can cause model instability. One solution is to use very high Manning's n values (.2 to 1.0) in the ineffective flow zones, so when they turn off the increase in conveyance is not so great. This is also more physically appropriate as the cross sections just upstream and downstream can not flow completely freely because of the bridge embankment.

When an isolated high ground area is causing an instability problem, the user must decide if this high ground is better modeled with the levee option or with ineffective flow.

Profile Summary Tables

- Sometimes visual clues are not available. Tabular output help.
  - lateral inflow/outflow
    - Tributaries
    - Interaction with storage areas
  - Lateral structure flow
  - Inline structure flow
  - Flow inconsistency
    - Main channel to overbanks
  - Other internal boundaries
    - Groundwater

Profile Output Table - Standard Table 1

File Options Std. Tables Locations Help

HEC-RAS Plan: Unstead lat River: Beaver Creek Reach: Kentwood — Reload Data

Reach	River Sta	Profile	Q Total (cfs)	Min Ch El (ft)	W.S. Elev (ft)	Crit W.S. (ft)
Kentwood	5.44	10FEB1999 2100	13417.00	203.90	215.45	214.10
Kentwood	5.425	10FEB1999 2100	13410.05	203.30	214.81	213.81
Kentwood	5.41	10FEB1999 2100	13406.17	202.70	214.12	212.39
Kentwood	5.4		Lat Struct			
Kentwood	5.39	10FEB1999 2100	13407.10	202.70	213.73	212.53
Kentwood	5.37	10FEB1999 2100	13409.02	202.72	213.59	213.03
Kentwood	5.35	10FEB1999 2100	10765.62	202.74	213.73	212.60
Kentwood	5.33	10FEB1999 2100	8942.68	202.76	213.78	212.06
Kentwood	5.31	10FEB1999 2100	8946.04	202.78	213.73	211.57
Kentwood	5.29	10FEB1999 2100	8949.46	202.80	213.70	210.86
Kentwood	5.274	10FEB1999 2100	8955.85	202.51	213.61	210.83
Kentwood	5.258	10FEB1999 2100	8964.30	202.22	213.52	210.87
Kentwood	5.242	10FEB1999 2100	8975.48	201.93	213.41	210.85
Kentwood	5.226	10FEB1999 2100	8990.36	201.64	213.30	210.92
Kentwood	5.21	10FEB1999 2100	9010.23	201.35	213.17	210.88
Kentwood	5.194	10FEB1999 2100	9036.77	201.06	213.04	210.83
Kentwood	5.178	10FEB1999 2100	9073.05	200.77	212.89	210.81
Kentwood	5.162	10FEB1999 2100	9122.71	200.48	212.73	210.77
Kentwood	5.146	10FEB1999 2100	9191.87	200.19	212.56	210.72

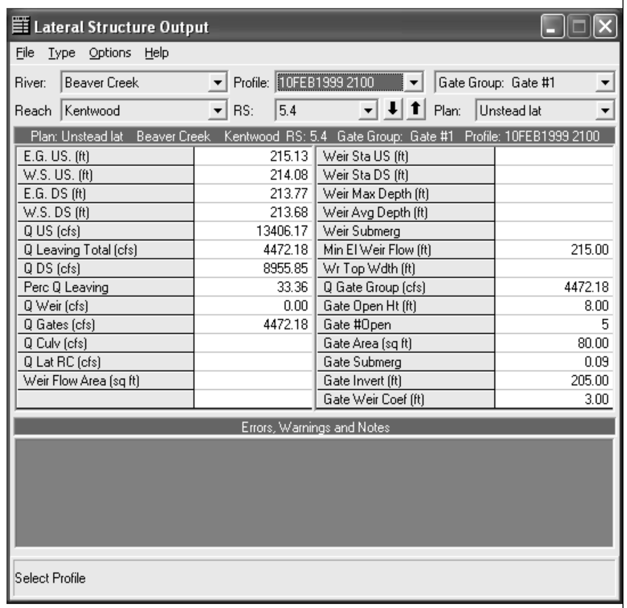
Total flow in cross section.

Hydrologic Engineering Center 41

Often times the graphical options alone are not adequate to determine the source of instability. Another option is to go to the profile output table and analyze values of hydraulic parameters from one cross section to the next or from one profile to the next. Problems that don't always show up graphically are lateral inflows and outflows, groundwater interaction and the effects of lateral structures. It is imperative that the important hydraulic parameters (flow, depth, area, storage) change as gradually as possible. Flow consistency between the overbanks and the main channel is also important.

Detailed Output Tables

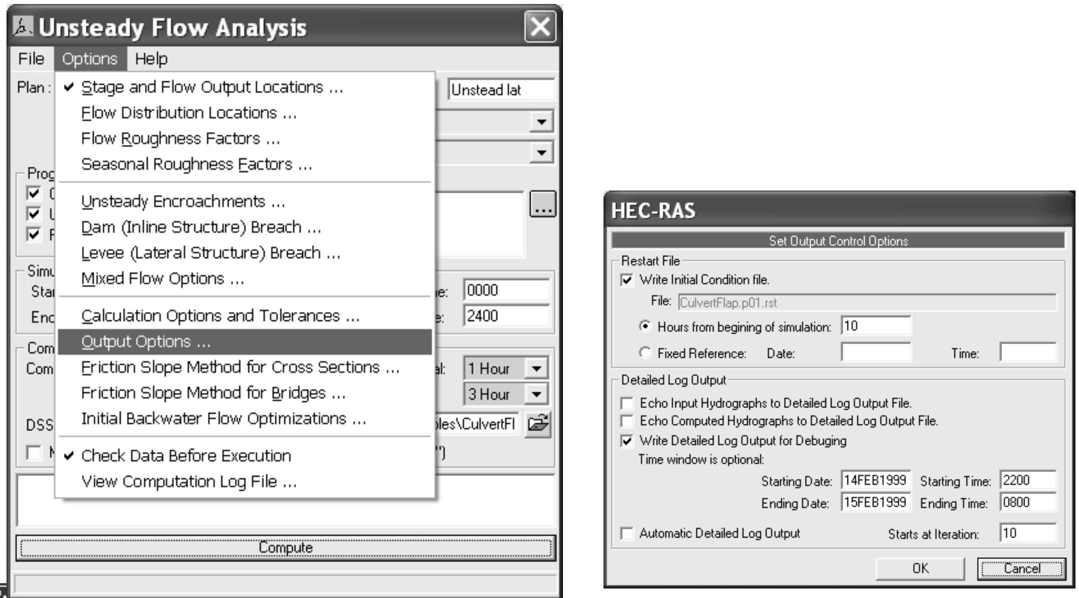
- Very good for looking at details of :
  - Inline Structures
  - Lateral Structures
  - Bridges/Culverts
  - Storage Areas
  - Pump Stations
  - Cross Sections



Hydrologic Engineering Center 42

Detailed output tables are available for many types of nodes in HEC-RAS, including: cross sections, bridges, culverts, inline structures, lateral structures, storage areas, and pump stations. These detailed tables can be very helpful in seeing what is going on at that structure for a particular time step.

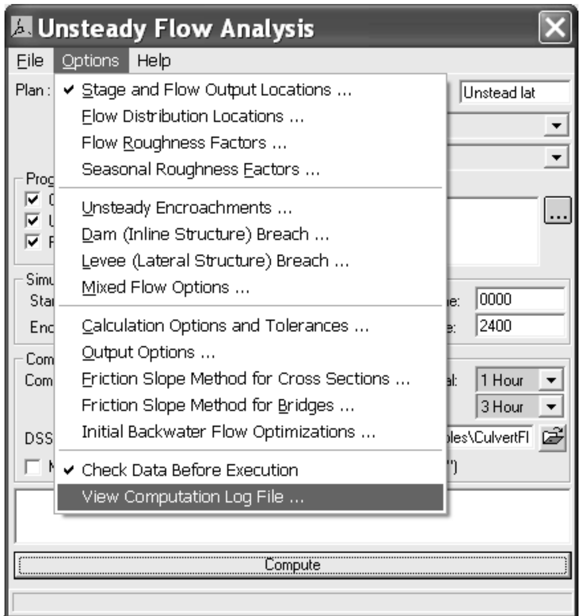
Turning on Detailed Log Output for Debugging



Hydrologic Engineering Center 43

As shown in the figure above, the section at the bottom half of this editor is used for controlling the **detailed log output**. Three check boxes are listed. The first box can be used to turn on an echo of the hydrograph input to the model. This can be used to ensure that the model is receiving the correct flow data. The second check box can be used to turn on an echo of the computed hydrographs that will be written to the HEC-DSS. This is a good option for checking what was computed. However, if the user has selected to have hydrographs computed at many locations, this could end up taking a lot of file and disk space. The third check box is used to control the detailed output of results from the unsteady flow simulation. Selecting this options will cause the software to write detailed information on a time step by time step basis. This option is useful when the unsteady flow simulation is going unstable or completely blowing up (stopping). Checking this box turns on the detailed output for every time step. The user has the option to limit this output to a specific time window during the unsteady flow simulation. Limiting the log output is accomplished by entering a starting date and time and an ending date and time.

Viewing Detailed Log Output



Viewing Detailed Log Output: After the user has turned on the detailed log output option, re-run the unsteady flow simulation. The user can then view the detailed log output by selecting **View Computational Log File** from the **Options** menu of the Unsteady flow simulation window. When this option is selected the detailed log output file will be loaded into the default text file viewer for your machine (normally the NotePad.exe program, unless you have changed this option within HEC-RAS).

What is found in the detailed Output

- DSS Data – shows all the data that was read from DSS.
- Unsteady Flow Computations Output – Detailed unsteady flow calculations:
  - Job control parameters
  - Initial conditions calculations
  - Detailed output for each time step
- TABLE Output – final hydrographs that are written to DSS

Hydrologic Engineering Center 45

The detailed log output file will contain the following output:

DSS Output: Shows all of the hydrograph data that will be used as input to the model, including data read from HEC-DSS.

Unsteady Flow Computations Output: Detailed unsteady flow calculations including:

Job control parameters

Initial conditions calculations

Detailed output for each time step

Table Output: Final computed hydrographs that are written to HEC-DSS.

Initial Conditions Output

Diamond.bco - Notepad

File Edit Format Help

Initial Conditions from Backwater

Diamond North

Riv. Sta.	Flow	WSEL	Crit Depth	EG Slope	Area	Topwidth	Velocity	Error	Converged
6.0	100.0	11.21		0.0000036	320.88	43.21	0.312	0.00000	T
5.8	100.0	11.20		0.0000027	355.63	44.00	0.281	0.00000	T
5.6	100.0	11.20		0.0000020	391.05	44.80	0.256	0.00000	T
5.4	100.0	11.20		0.0000015	443.55	100.40	0.225	0.00000	T
5.2	100.0	11.20		0.0000011	553.25	174.29	0.181	0.00000	T
5.0	100.0	11.20		0.0000008	720.29	230.22	0.139	0.00000	T
4.8	100.0	11.20		0.0000008	720.08	230.22	0.139	0.00000	T
4.6	100.0	11.20		0.0000008	719.88	230.22	0.139	0.00000	T
4.4	100.0	11.19		0.0000008	719.68	230.22	0.139	0.00000	T
4.2	100.0	11.19		0.0000008	719.47	230.21	0.139	0.00000	T
4.0	100.0	11.19		0.0000008	719.27	230.21	0.139		

Diamond Northwest

Riv. Sta.	Flow	WSEL	Crit Depth	EG Slope	Area	Topwidth	Velocity	Error	Converged
4.0	70.0	11.19		0.0000004	719.32	230.21	0.097	0.00000	T
3.8	70.0	11.19		0.0000004	719.24	230.21	0.097	0.00000	T
3.6	70.0	11.19		0.0000004	719.16	230.21	0.097	0.00000	T
3.4	70.0	11.19	-0.51	0.0000004	498.62	45.00	0.140	0.00000	T
3.395 Culvert									
3.39	70.0	11.00	-0.51	0.0000005	489.99	45.00	0.143	0.00000	T
3.35	70.0	11.00		0.0000005	489.99	45.00	0.143		

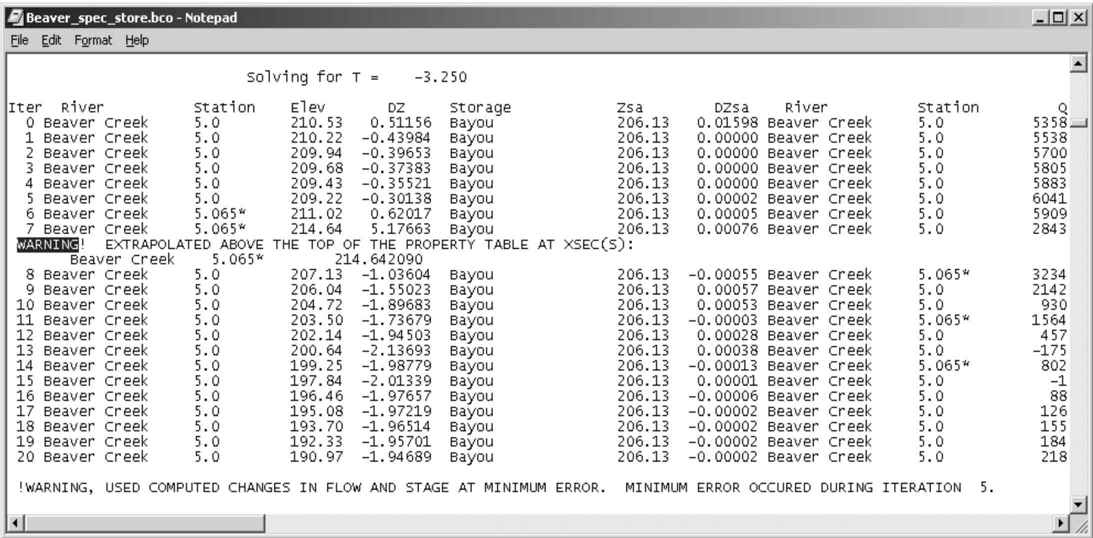
Diamond Northeast

Riv. Sta.	Flow	WSEL	Crit Depth	EG Slope	Area	Topwidth	Velocity	Error	Converged
3.9999	30.0	11.00		0.0000001	675.21	230.00	0.044	0.00000	T
3.77768	30.0	11.00		0.0000001	675.19	230.00	0.044	0.00000	T
3.55547	30.0	11.00		0.0000001	675.17	230.00	0.044	0.00000	T
3.33326	30.0	11.00		0.0000001	675.14	230.00	0.044	0.00000	T
3.11105	30.0	11.00		0.0000001	675.12	230.00	0.044	0.00000	T
2.88884	30.0	11.00		0.0000001	675.10	230.00	0.044	0.00000	T
2.66663	30.0	11.00		0.0000001	675.08	230.00	0.044	0.00000	T
2.44442	30.0	11.00		0.0000001	675.06	230.00	0.044	0.00000	T
2.22221	30.0	11.00		0.0000001	675.03	230.00	0.044	0.00000	T

46

The program lists the computed initial conditions from a backwater calculation for each of the river/reaches. They are generally listed in an upstream to downstream order. However, they are computed from downstream to upstream under the assumption of subcritical flow.

Example Detailed Time Step Output for cross sections



Beaver_spec_store.bco - Notepad

File Edit Format Help

```
                          Solving for T =    -3.250

Iter  River          Station    Elev      DZ     Storage          Zsa       DZsa    River          Station       Q
  0 Beaver Creek     5.0        210.53   0.51156 Bayou            206.13   0.01598 Beaver Creek    5.0        5358
  1 Beaver Creek     5.0        210.22  -0.43984 Bayou            206.13   0.00000 Beaver Creek    5.0        5538
  2 Beaver Creek     5.0        209.94  -0.39653 Bayou            206.13   0.00000 Beaver Creek    5.0        5700
  3 Beaver Creek     5.0        209.68  -0.37383 Bayou            206.13   0.00000 Beaver Creek    5.0        5805
  4 Beaver Creek     5.0        209.43  -0.35521 Bayou            206.13   0.00000 Beaver Creek    5.0        5883
  5 Beaver Creek     5.0        209.22  -0.30138 Bayou            206.13   0.00002 Beaver Creek    5.0        6041
  6 Beaver Creek     5.065*     211.02   0.62017 Bayou            206.13   0.00005 Beaver Creek    5.0        5909
  7 Beaver Creek     5.065*     214.64   5.17663 Bayou            206.13   0.00076 Beaver Creek    5.0        2843
 WARNING!  EXTRAPOLATED ABOVE THE TOP OF THE PROPERTY TABLE AT XSEC(S):
       Beaver Creek    5.065*        214.642090
  8 Beaver Creek     5.0        207.13  -1.03604 Bayou            206.13  -0.00055 Beaver Creek    5.065*     3234
  9 Beaver Creek     5.0        206.04  -1.55023 Bayou            206.13   0.00057 Beaver Creek    5.0        2142
 10 Beaver Creek     5.0        204.72  -1.89683 Bayou            206.13   0.00053 Beaver Creek    5.0         930
 11 Beaver Creek     5.0        203.50  -1.73679 Bayou            206.13  -0.00003 Beaver Creek    5.065*     1564
 12 Beaver Creek     5.0        202.14  -1.94503 Bayou            206.13   0.00028 Beaver Creek    5.0         457
 13 Beaver Creek     5.0        200.64  -2.13693 Bayou            206.13   0.00038 Beaver Creek    5.0        -175
 14 Beaver Creek     5.0        199.25  -1.98779 Bayou            206.13  -0.00013 Beaver Creek    5.065*      802
 15 Beaver Creek     5.0        197.84  -2.01339 Bayou            206.13   0.00001 Beaver Creek    5.0          -1
 16 Beaver Creek     5.0        196.46  -1.97657 Bayou            206.13  -0.00006 Beaver Creek    5.0          88
 17 Beaver Creek     5.0        195.08  -1.97219 Bayou            206.13  -0.00002 Beaver Creek    5.0         126
 18 Beaver Creek     5.0        193.70  -1.96514 Bayou            206.13  -0.00002 Beaver Creek    5.0         155
 19 Beaver Creek     5.0        192.33  -1.95701 Bayou            206.13  -0.00002 Beaver Creek    5.0         184
 20 Beaver Creek     5.0        190.97  -1.94689 Bayou            206.13  -0.00002 Beaver Creek    5.0         218

 !WARNING, USED COMPUTED CHANGES IN FLOW AND STAGE AT MINIMUM ERROR.  MINIMUM ERROR OCCURED DURING ITERATION  5.
```

Hydrologic Engineering Center 47

One way to find and locate potential stability problems with the solution is to do a search in the file for the word "**WARNING**". The user then needs to look at the detailed output closely to try and detect both where and why the solution is going bad. The variables that are printed out during the iterations are the following:

Iter	= Iteration Number.
River	= River Name of the location with the largest error in stage.
Station	= River station with the larges error in the calculated stage.
ELEV	= Computed water surface elevation at that river station.
DZ	= The "Numerical Error" in the computed stage at that location.
Storage	= Name of the storage area that has the larges error for this iteration.
Zsa	= Computed elevation of the storage area.
Dzsa	= The "Numerical Error" in the computed storage area elevation.
River	= River Name of the location with the largest error in flow.
Station	= River station with the largest error in the calculation of flow.
Q	= Computed flow
DQ	= The "Numerical Error" in the computed flow at the listed river station

Note: If the program goes to the maximum number of iterations, it will choose the iteration that had the minimum amount of error, set that as the solution for the current time step, and then go to the next time step.

Example Detailed Time Step Output for cross sections - Continued

beaver.bco - Notepad

File Edit Format Help

```
     COMPUTED STAGES AND DISCHARGES AT T =      0.1167 HOURS -  2/10/1999 AT 0007 HOURS

             Beaver Creek     Kentwood

Riv. Station     Z        Q      V       Riv. Station     Z        Q      V       Riv. Station     Z        Q      V
    5.99     213.03    599.   1.09           5.97     212.94    588.   1.22          5.951     212.83    579.   1.37
    5.93     212.71    571.   1.56           5.913    212.56    564.   1.79          5.894     212.38    558.   2.04
   5.875     212.19    552.   2.34           5.855    211.98    547.   2.65          5.836     211.74    543.   2.91
    5.81     211.52    540.   2.96           5.798    211.36    536.   2.45          5.779     211.24    532.   1.76
    5.76     211.17    528.   1.18           5.741    211.07    523.   1.64           5.72     210.91    521.   2.15
   5.703     210.75    519.   2.31           5.685    210.61    517.   2.30          5.666     210.48    516.   2.19
   5.647     210.37    515.   1.92           5.628    210.27    514.   1.62           5.61     210.21    513.   1.31
   5.593     210.13    512.   1.52           5.576    210.03    511.   1.80          5.559     209.93    511.   2.06
   5.542     209.85    510.   2.23           5.525    209.78    510.   2.24          5.508     209.72    510.   2.14
   5.491     209.67    510.   1.94           5.474    209.64    510.   1.76          5.457     209.61    510.   1.60
    5.44     209.58    510.   1.47           5.425    209.58    509.   1.11           5.41     209.57    509.   0.88
    5.39     209.54    509.   0.88            5.37    209.52    509.   1.11           5.35     209.48    509.   1.46
    5.33     209.40    510.   1.67            5.31    209.28    510.   1.67           5.29     209.15    510.   1.47
   5.274     208.95    510.   1.66           5.258    208.68    511.   1.95          5.242     208.29    511.   2.55
   5.226     207.85    511.   3.05            5.21    207.46    512.   3.29          5.194     207.15    512.   3.13
   5.178     206.95    513.   2.68           5.162    206.83    513.   2.16          5.146     206.75    514.   1.75
    5.13     206.71    514.   1.45           5.113    206.65    515.   1.50          5.097     206.59    515.   1.56
   5.081     206.53    516.   1.60           5.065    206.46    517.   1.63          5.048     206.39    517.   1.65
   5.032     206.31    518.   1.66           5.016    206.23    519.   1.66            5.0     206.13    519.   1.64


                           Solving for T =     0.133

Iter  River          Station     Elev      DZ       River          Station        Q          DQ
  0 Beaver Creek     5.99        213.07   0.03530  Beaver Creek    5.99           613        14
  1 Beaver Creek     5.0         206.13  -0.00050  Beaver Creek    5.93           584         1
```

Hydrologic Engineering Center 48

During the unsteady flow computations, the program will output detailed information for cross sections, bridges/culverts, inline weir/spillways, lateral weir/spillways, storage areas, and storage area connections. This information should be reviewed closely when the software is having stability problems. An example of the detailed output for cross sections is shown in the Figure above.